Tutorial 1

Helius:MCT™ Version 2.0 for ANSYS
July, 2009

Abstract

This document provides a step-by-step tutorial that demonstrates the use of Helius:MCT. The primary emphasis is the creation of ANSYS models that are compatible with Helius:MCT and the viewing of special solution variables that are computed by Helius:MCT. Tutorial 1 demonstrates the use of ANSYS Mechanical in building an ANSYS finite element model.

For questions, comments or further information, contact Firehole Technologies at support@fireholetech.com

Legal Notices

Copyright 2009, Firehole Technologies, Inc.

Helius:MCT is a trademark of Firehole Technologies, Inc. Any use of the Helius:MCT trademark requires the prior written consent of Firehole Technologies, Inc.

ANSYS Mechanical is a trademark of ANSYS, Inc.

FIREHOLE TECHNOLOGIES
ADVANCED COMPOSITES ANALYSIS

Table of Contents

1 INTRODUCTION 3

2 TUTORIAL STEPS 4

2.1 CREATING THE PART 4
2.2 DEFINE AN ELEMENT TYPE 5
2.3 CREATING A USER MATERIAL WITH HELIUS:MCT 5
2.4 MESH THE PART 7
2.5 DEFINING A COMPOSITE LAYUP 8
2.6 DEFINE AND ASSIGN THE ELEMENT COORDINATE SYSTEM 9
2.7 APPLYING BOUNDARY CONDITIONS 10
2.8 DEFINING THE LOAD 11
2.9 SPECIFY SOLUTION CONTROLS 12
2.10 DEFINING SOLUTION OUTPUT TO THE RESULTS FILE 13
2.11 SOLVE THE MODEL 13
2.12 VIEWING AND INTERPRETING THE RESULTS 13

3 SUMMARY 16

Table of Figures

Figure 1: Dimensions and loading of composite plate 3
Figure 2: Helius:MCT GUI 7
Figure 3: Plate mesh 8
Figure 4: Create and Modify Shell Sections dialog box 9
Figure 5: Local coordinate system 10
Figure 6: Location (red) of bottom surface boundary condition 11
Figure 7: Location (red) of top surface load boundary condition 12
Figure 8: PLESOL contour plot of ply 1 at the end of the step 14
Figure 9: PLESOL contour plot of ply 2 at the end of the step 15

Helius:MCT Tutorial 1

Note: The following tutorial is intended to be used with ANSYS Mechanical Version 12. As a result, some of the results may be different if previous versions are used.

1 Introduction

The following tutorial provides step-by-step instructions to create and analyze a simple composite plate using the Helius:MCT GUI in ANSYS Mechanical™. The problem consists of a flat composite plate with a hole in the center subject to fixed boundary conditions on one end, and a displacement controlled load of 0.12" (0.8% of the plate length) on the other end. The plate is made of IM7/8552, the layup is $[0/\pm45/90]_s$, and the ply thickness is 0.005" which results in a plate thickness of 0.04". The seed size for the mesh is 0.2, and 8-node layered solid shell elements (SOLSH190) are used. The dimensions, boundary conditions, and load are shown in Figure 1. After the model is created and the finite element analysis is completed, steps are provided to view and interpret the results.

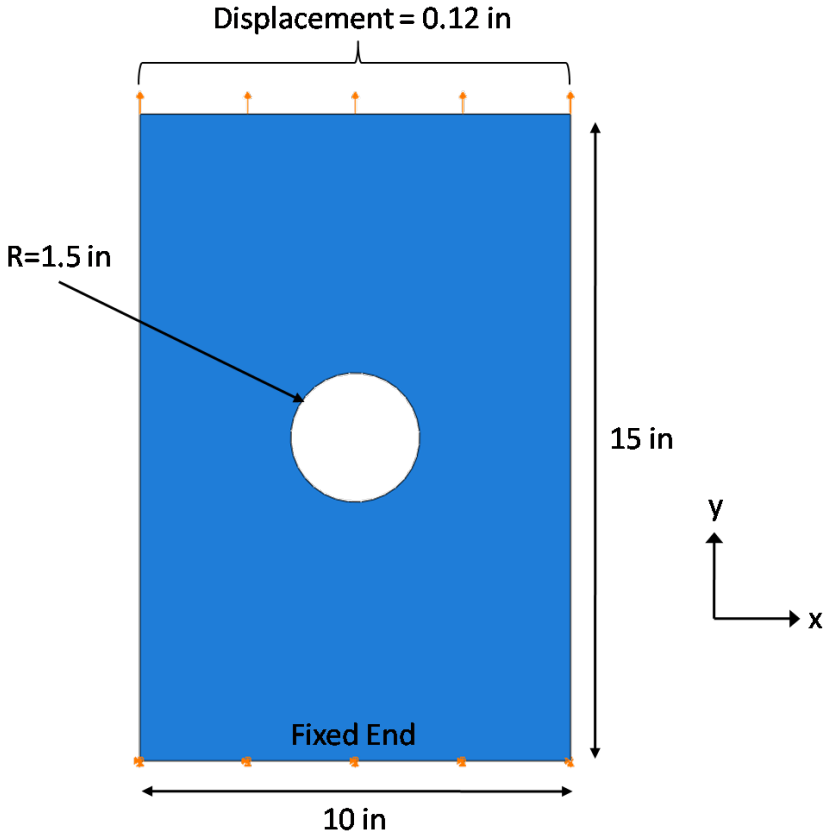

Figure 1: Dimensions and loading of composite plate

If one is interested only in learning how to interpret Helius:MCT's results, an ANSYS input file (HeliusMCT_Tutorial_1_ANSYS_v2.inp) is available for download from Firehole's internet based User Portal that can be used to generate an ANSYS results (*.rst) file. In this case, run the input file and skip to Section 2.12. Before submitting the file for analysis, ensure that the matID values in the input file match

the matID for IM7_8552 on your workstation. The matID for IM7_8552 can be determined by selecting IM7_8552 in the Helius GUI within Ansys as detailed in Section 2.3.

2 Tutorial Steps

In the following tutorial, elementary modeling details are omitted as it is assumed that the user has previous experience in the ANSYS Mechanical environment. For example, it is assumed that the user knows how to enter a command into the command prompt so that when an instruction tells the user to enter text into the command prompt, the instructions to perform that task are omitted. Please refer to the ANSYS documentation before completing this tutorial if you are unfamiliar with the ANSYS GUI.

2.1 Creating the part

Defining the part geometry is generally the first step in the development of a finite element model. Here, the plate geometry is defined to generate a 3-dimensional part.

1. Open ANSYS 12.0.
2. Click **Main Menu→Preprocessor→Modeling→Create→Volumes→Block→By 2 Corners & Z**.
3. In the dialog box that appears, enter the following values:
    a. **WP X = 0**
    b. **WP Y = 0**
    c. **Width = 10**
    d. **Height = 15**
    e. **Depth = 0.04**
4. Click **OK**.
5. Click **Main Menu→Preprocessor→Modeling→Create→Volumes→Cylinder→Solid Cylinder**.
6. In the dialog box that appears, enter the following values:
    a. **WP X = 5**
    b. **WP Y = 7.5**
    c. **Radius = 1.5**
    d. **Depth = 0.04**
7. Click **OK**.
8. Click **Main Menu→Preprocessor→Modeling→Operate→Booleans→Subtract→Volumes**.
9. Select the block from the Graphics Window. A message will appear noting that there are two volumes at this location. Make sure that the Volume 1 is the Picked Volume and click **OK**.
10. Click **OK** in the Subtract Volumes dialog box.

11. Select the cylinder from the Graphics Window. Make sure that Volume 2 is the Picked Volume and click **OK**.
12. Click **OK** in the Subtract Volumes dialog box. The plate geometry is now defined.

2.2 Define an element type

There are several ANSYS elements that have layered functionality. For this tutorial, element type SOLSH190 is used. SOLSH190 is a 3-dimensional, 8-node layered solid shell. This element is commonly used to model plate-like structures.

1. Click **Main Menu→Preprocessor→Element Type→Add/Edit/Delete**.
2. In the dialog box that appears, click **Add**.
3. Select **Solid-Shell** in the Library of Element Types dialog box. Click **OK**.
4. Click **Close** in the Element Types dialog box.

In order to store top and bottom section point data for all layers within each element, KEYOPT(8) is used.

5. Type **KEYOPT,1,8,1** into the command prompt and press **Enter**.

2.3 Creating a user material with Helius:MCT

The Helius:MCT GUI is the central interface between the user and Helius:MCT. It allows the user to choose from a variety of material and analysis options including the following:

- Choice of material
- 4 unit systems
- Principal material direction
- Progressive failure analysis
- Pressure induced strength enhancement
- Pre-failure and post-failure nonlinearity
- Matrix and fiber post-failure stiffness fractions
- Output of fiber and matrix stresses and strains to the results file

Using these options, the user can tailor his or her analysis to the requirements of the problem. For a detailed discussion of the options available, refer to Section 3.1 of the User's Manual.

In the following steps, a user-material is created for the plate, and progressive failure analysis is requested using the Helius:MCT GUI.

1. Click on the **HELIUS** button from the ANSYS Toolbar. The Helius:MCT 2.0 For ANSYS dialog box appears.

   Note that the HELIUS button only works in the model creation preprocessor (/PREP7).
2. From the Material Library list, select the material **IM7_8552**.

The unit dependent Engineering Constants specific to this material are listed in the dialog box for the user to review. Note that the ANSYS Material ID (matID) that corresponds to this material is also shown in the dialog box. This value may vary from computer to computer. It is useful to know the Material ID when working with input files.

3. Since this model uses inches and pounds as base units, select **lb/in/R** from the Select Model Units list.

   There are 4 unit systems to choose from. The default unit system is N/m/K.

4. Select **1** as the fiber direction.

   2 can also be used as the fiber direction, but it would require a different composite layup orientation than the 1 direction. As a general rule, it is recommended that 1 be used as the fiber direction to maintain consistency from model to model. On occasion, however, it will not be possible to create an orientation in ANSYS that allows for the 1 direction to be the fiber direction due to the combination of complex model geometry and orientation limitations. In such cases, it may be necessary to use the 2 direction as the fiber direction.

5. Check the Progressive Failure Analysis box.

   In this tutorial, only progressive failure is included in the analysis because it is the foundation for Helius:MCT's nonlinear multiscale constitutive model.

6. Set the Matrix Post-Failure Stiffness value to **0.10** and the Fiber Post-Failure Stiffness value to **0.01**. The dialog box should appear as shown in Figure 2.

   These values specify the ratio of damaged to undamaged material properties for the material. For example, a value of 0.01 for the Fiber Post-Failure Stiffness means that when an integration point undergoes a fiber failure event, the material properties at that integration point will degrade to 1% of the undamaged properties.

7. Click **OK**.

8. For Ansys 11 users, enter the following into the command prompt, ensuring that *matID* is replaced with the value that corresponds to IM7_8552:

   MP, EX, *matID*, 1e7

   The above command defines a longitudinal elastic modulus definition for the Helius:MCT material. This value is used to fill a place holder required by ANSYS when using layered elements with a user-defined material. The value of EX does not affect the results. This value is not required in Ansys 12.

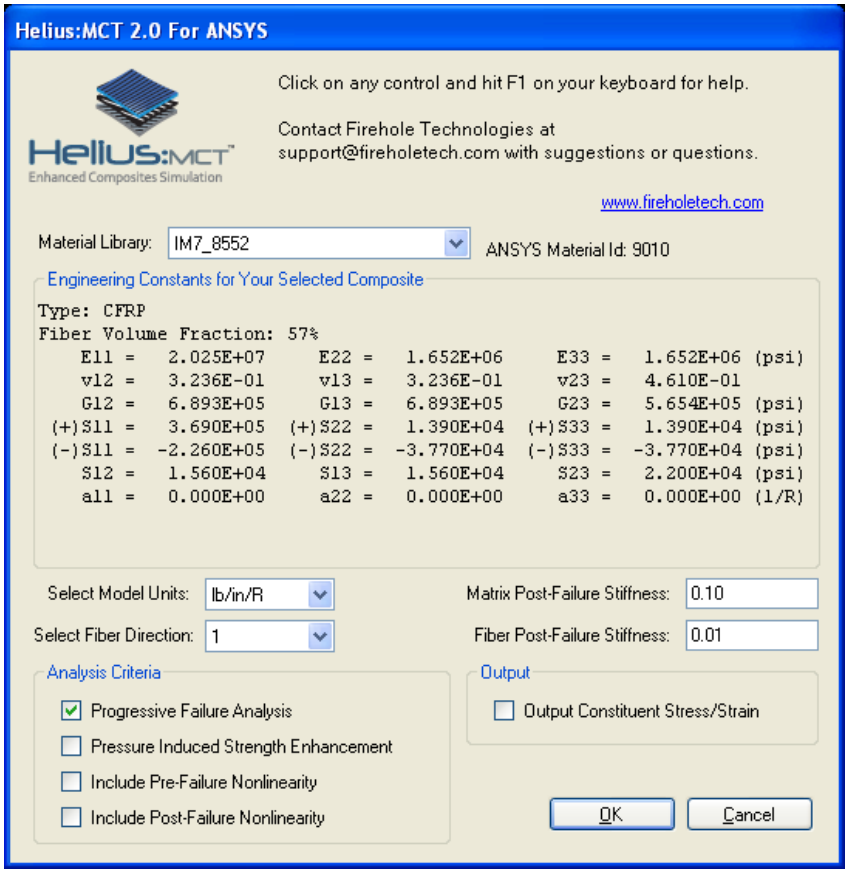
Helius:MCT 2.0 For ANSYS
Helius:MCT
Enhanced Composites Simulation
Click on any control and hit F1 on your keyboard for help.
Contact Firehole Technologies at
support@fireholetech.com with suggestions or questions.
www.fireholetech.com
Material Library: IM7_8552
ANSYS Material Id: 9010
Engineering Constants for Your Selected Composite
Type: CFRP
Fiber Volume Fraction: 57%
E11 = 2.025E+07 E22 = 1.652E+06 E33 = 1.652E+06 (psi)
v12 = 3.236E-01 v13 = 3.236E-01 v23 = 4.610E-01
G12 = 6.893E+05 G13 = 6.893E+05 G23 = 5.654E+05 (psi)
(+)S11 = 3.690E+05 (+)S22 = 1.390E+04 (+)S33 = 1.390E+04 (psi)
(-)S11 = -2.260E+05 (-)S22 = -3.770E+04 (-)S33 = -3.770E+04 (psi)
S12 = 1.560E+04 S13 = 1.560E+04 S23 = 2.200E+04 (psi)
all = 0.000E+00 a22 = 0.000E+00 a33 = 0.000E+00 (1/R)
Select Model Units: lb/in/R
Select Fiber Direction: 1
Matrix Post-Failure Stiffness: 0.10
Fiber Post-Failure Stiffness: 0.01
Analysis Criteria
Progressive Failure Analysis
Pressure Induced Strength Enhancement
Include Pre-Failure Nonlinearity
Include Post-Failure Nonlinearity
Output
Output Constituent Stress/Strain
OK
Cancel

Figure 2: Helius:MCT GUI

2.4 Mesh the part

In meshing the composite plate, a global seed size of 0.2 is used to ensure that the mesh is fine enough to capture the location and propagation of failure, but also coarse enough for a short solution time.

1. Click **Main Menu→Preprocessor→Meshing→Mesh Tool**.
2. To set the global seed size, click the **Set** button next to the Global Size Controls, enter **0.2** in the **SIZE** entry box, and click **O**K.
3. Select the **Hex** and **Sweep** radio buttons and click the **Sweep** button.
4. Select the part from the Graphics Window and click **OK** in the Volume Sweeping dialog box. The part is now meshed and should be similar to the mesh shown in Figure 3.

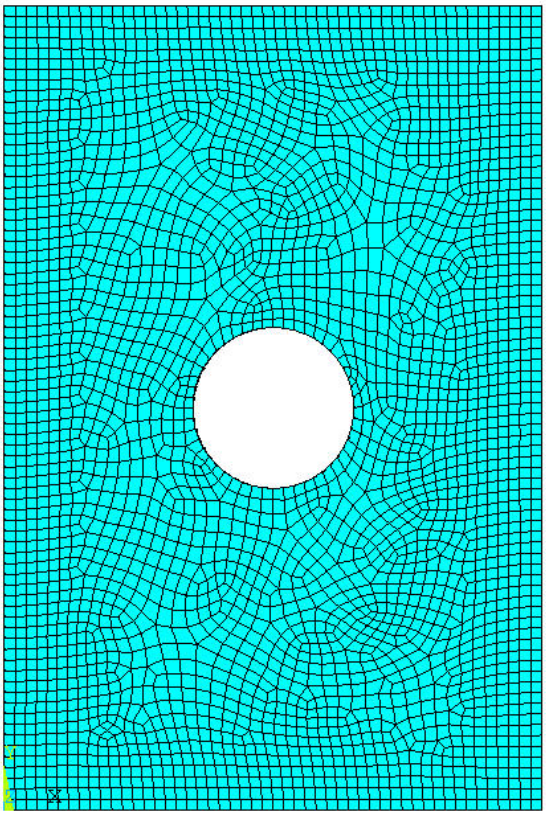

Figure 3: Plate mesh

2.5 Defining a composite layup

After the part and the material have been created, a composite layup section can be created. The composite layup editor is used to create plies and to assign materials and orientations to these plies. In this step, a composite layup that represents the plate layup is created and defined.

1. Select **Main Menu→Preprocessor→Sections→Shell→Lay-up→Add/Edit**.
2. Since the plate is eight plies thick, click the **Add** Layer button until there are **eight layers**.
3. Enter **0.005** for all thickness values.
4. Enter the orientations for plies 1 thru 8 as **0, 45, -45, 90, 90, -45, 45, 0**, respectively. The dialog box should appear as shown in Figure 4. Note that the Material ID values may differ depending on the materials in your Material Library.
5. Click **OK**.

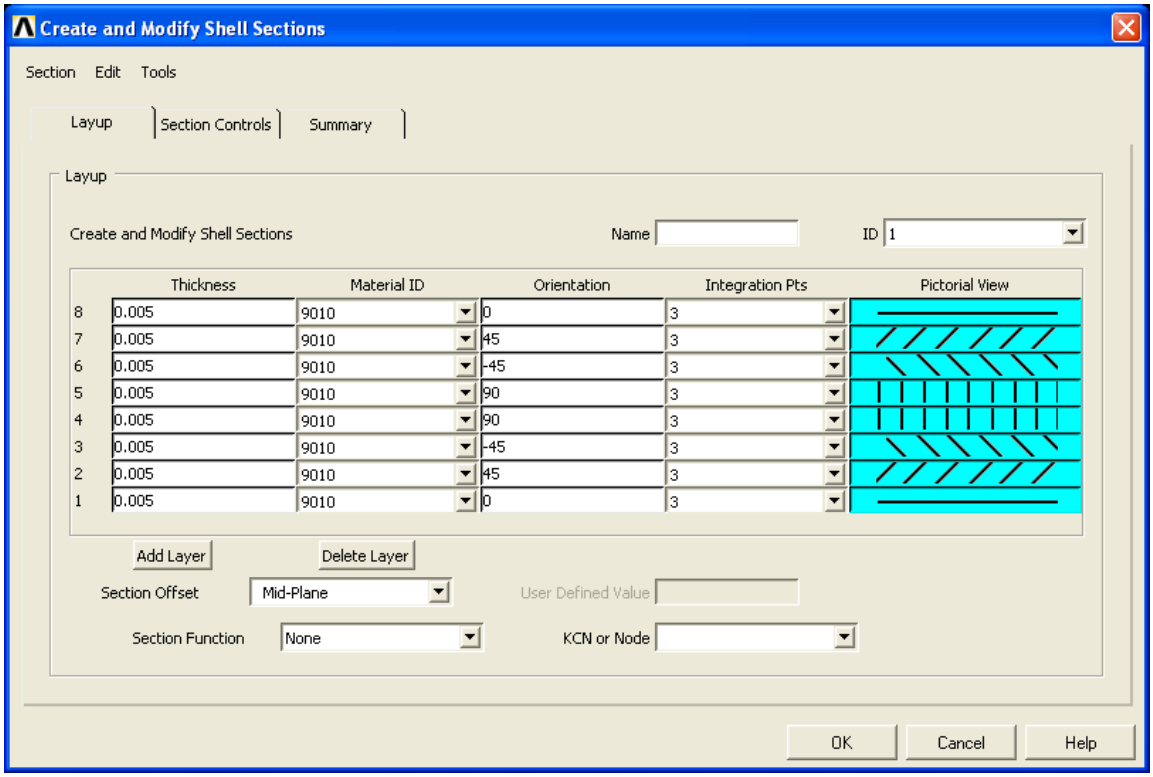

Figure 4: Create and Modify Shell Sections dialog box

2.6 Define and assign the element coordinate system

In order to align the 0 degree plies with the loading direction, a local coordinate system needs to be defined and assigned to the elements.

1. To define a local, rectangular coordinate system with the x-axis aligned with the global loading direction (the global y-axis), enter the following into the command prompt:

   LOCAL,11,CART,0,0,0,90,0,0

2. To assign the local coordinate system to the elements, first select the elements by clicking **Utility Menu→Select→Everything**, then enter **ESYS,11** into the command prompt. Figure 5 shows the local coordinate system relative to the base of the part.

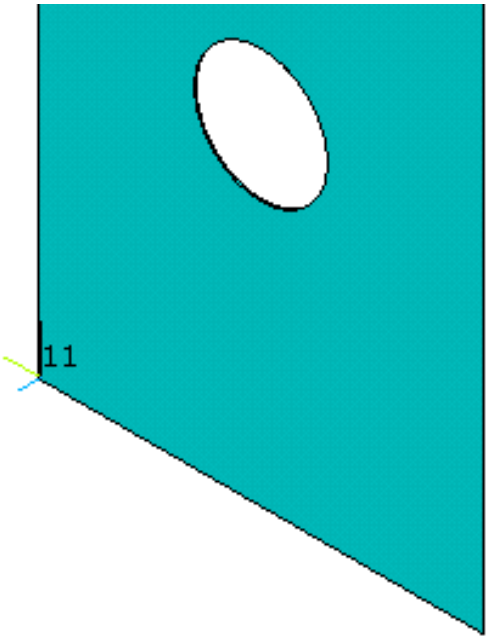

Figure 5: Local coordinate system

2.7 Applying boundary conditions

The following steps create a boundary condition that fixes the bottom surface of the plate:

1. Select **Preprocessor→Loads→Define Loads→Apply→Structural→Displacements→On Areas**.
2. From the Graphics Window, select the bottom surface of the plate, as highlighted in red in Figure 6, and click **OK** in the Apply U,ROT on Areas dialog box.
3. In the dialog box that appears, enter a Displacement value of **0** for **All DOF** and click **OK**.

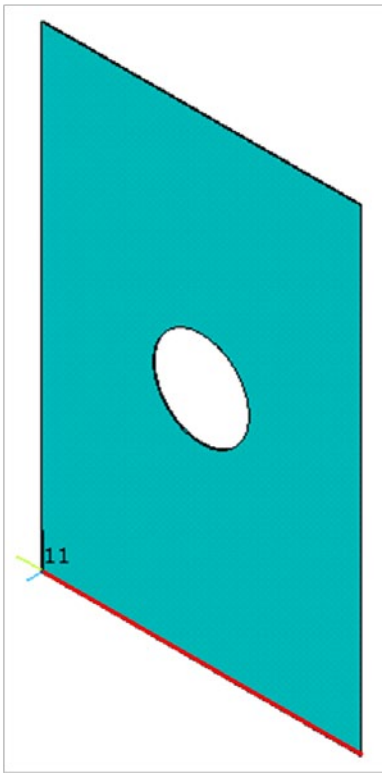

Figure 6: Location (red) of bottom surface boundary condition

2.8 Defining the load

A second boundary condition is created to impose a vertical displacement along the top surface of the plate. The plate is loaded by imposed displacements because it results in a much more gradual failure process than a comparable loading by applied forces. When a simple structure, such as this composite plate, begins to fail under the action of applied forces, the structure fails very rapidly because the load continues to increase as the load carrying capacity of the structure decreases. With displacement controlled loading, the load carried by the structure decreases as the structure fails which allows for a slower rate of failure.

1. Select **Preprocessor→Loads→Define Loads→Apply→Structural→Displacements→On Areas**.
2. From the Graphics Window, select the top surface of the plate, as highlighted in red in Figure 7, and click **OK** in the Apply U,ROT on Areas dialog box.
3. In the dialog box that appears, enter a Displacement value of **0.12** for **UY** and click **OK**.

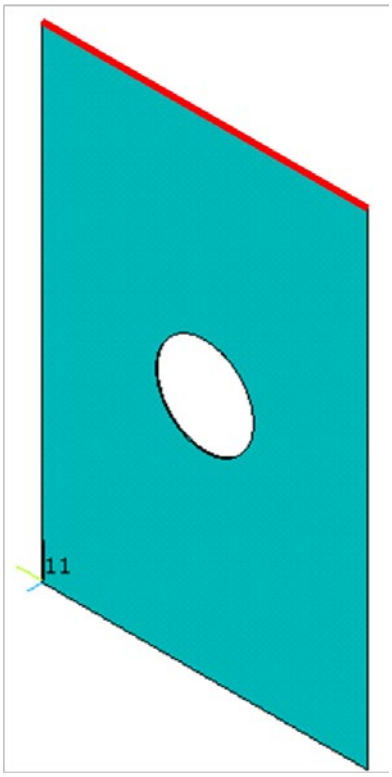

Figure 7: Location (red) of top surface load boundary condition

2.9 Specify solution controls

The default solution controls in ANSYS typically do not allow for a robust and efficient solution in a progressive failure simulation of a composite structure. There are specific solution control parameters that, when used in conjunction with Helius:MCT, allow for a solution with a decreased tendency for time increment cutbacks and faster overall solution time. These solution controls are defined in the following steps. For further information regarding each command, refer to the ANSYS documentation and the Helius:MCT User's Guide.

1. Enter the following commands into the command prompt:
    a. **NROPT,FULL,,OFF** – This command instructs ANSYS to use the 'Full' Newton Raphson algorithm and prevents ANSYS from using the 'Adaptive Descent' algorithm.
    b. **PRED,OFF,,OFF** – This command prevents ANSYS from using the converged solution at the last substep to estimate the solution for the current substep.
    c. **NSUBST,10,10,10** – This command is used to specify the number of substeps to be used in the analysis. Since the step time is 1, each substep time increment will be 0.1.
    d. **NEQIT,1000** – This command specifies the number of equilibrium iterations that must be performed before ANSYS evaluates the need to reduce the time increment size. It is intentionally large in order to force ANSYS to converge at each of the specified time steps.
    e. **TIME,1** – Specifies a step size equal to 1.
2. Switch to the solution processor by clicking **Main Menu→Solution**. Enter the following command into the command prompt:

a. **CNVTOL,F,,,0** – This command is used to define the convergence tolerance for residual node forces.

2.10 Defining solution output to the results file

In order to view the fiber and matrix failure states generated by Helius:MCT, the user must request that the state variables (SVAR) be written in the ANSYS results (rst) file.

1. Enter **OUTRES,ESOL,NONE** in the command prompt. This command suppresses writing all element solutions to the results file and is used to decrease the size of the results file. Depending on the model size, writing all element solutions to the results file will slow down the solution and can create very large files.
2. Enter **OUTRES,SVAR,ALL** in the command prompt. This command specifies that all state variables are to be written to the results file at every substep.
3. Enter **OUTRES,STRS,ALL** in the command prompt if you wish to view element nodal stresses.

2.11 Solve the model

The model is now defined and ready to be solved. To solve the model, click **Main Menu→Solution→Solve→Current LS**.

2.12 Viewing and interpreting the results

Helius:MCT generates several state variable outputs. In this step, a key variable (SVAR1) is viewed and discussed. SVAR1 is the state variable that keeps track of fiber and matrix failure within an element.

1. After the solution has completed, click **Main Menu→General Postproc**.
2. To view the true deformation scale, click **Utility Menu→PlotCtrls→Style→Displacement Scaling**. Select the **1.0 (true scale)** radio button in the dialog box that appears and click **OK**.

State variable SVAR1 is used to identify the discrete damage state of the composite material. To plot this variable, either the PLESOL or the PLNSOL commands may be used. The PLESOL contour plot does not average values across element boundaries while the PLNSOL contour plot does average vales across element boundaries. Since SVAR1 has discrete values, the PLESOL plot is the most useful.

3. To plot SVAR1 using a PLESOL contour plot, enter **PLESOL,SVAR,1** into the command prompt. The default plot will plot the bottom section point of the bottom layer and the top section point of the top layer.

4. Since there are 3 possible values of SVAR1 in this Tutorial, it is useful to adjust the contour color values. Click **Utility Menu→PlotCtrls→Style→Contours→Non-uniform Contours**. In the dialog box that appears, set **V1 = 1**, **V2 = 2**, and **V3 = 3** and click **OK**. The contour plot should be similar to the plot shown in Figure 8. Minor differences are usually the result of mesh variations.

The following table lists the three possible values for SVAR1 and the corresponding failure states and contour colors.

Value of SVAR1	Failure State	Color
1	No failure	Blue
2	Matrix failure	Green
3	Fiber and matrix failure	Red

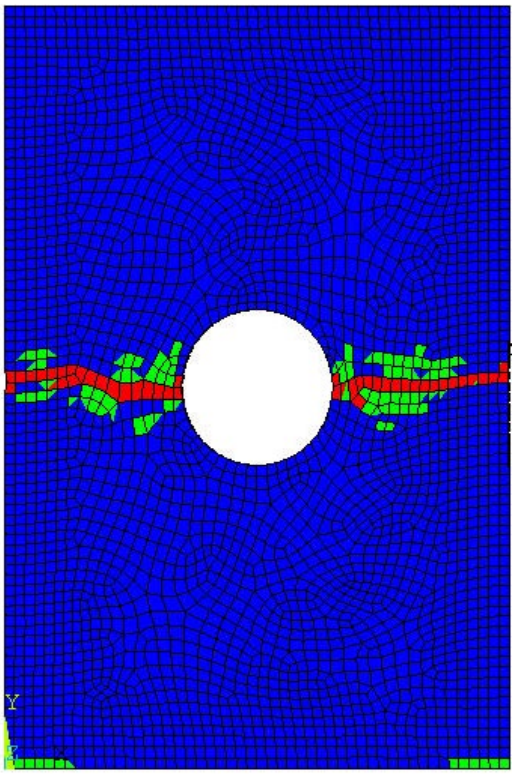

Figure 8: PLESOL contour plot of ply 1 at the end of the step

To view results in a specific ply (or layer) of the composite plate, the LAYER command can be used.

5. Enter **LAYER,2** in the command prompt to view results in ply 2 (90 degree ply). Note that it may be necessary to click **Utility Menu→Plot→Replot** to refresh the Graphics Window. The results for ply 2 are shown in Figure 9.

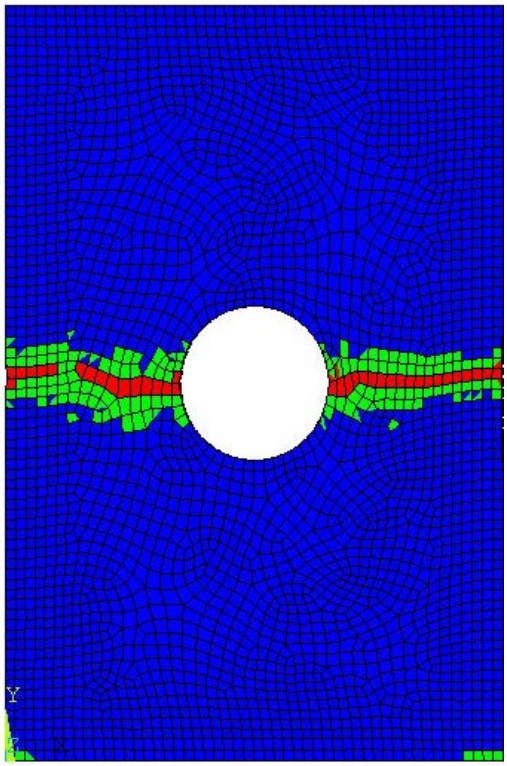

Figure 9: PLESOL contour plot of ply 2 at the end of the step

Viewing the progression of failure is often useful for visualizing the way a structure fails. Commands to view results at specific points in time can be accessed from **Main Menu→General Postproc→Read Results** or using the **SET** command.

6. Starting at Substep 1, progress through the step while refreshing the Graphics Window to determine when failure initiates and how failure propagates in ply 2.
    a. Enter the commands, **SET,1,*substep#*** or
    b. **Main Menu→General Postproc→Read Results→By Load Step**.
7. Starting at Substep 1, progress through the step while watching the viewport to determine when failure initiates and how failure propagates in ply 2.
    a. First matrix failure should occur at Time = 0.5 (substep 5).
    b. First fiber failure should occur at Time = 0.8 (substep 8).

3 Summary

A composite plate with a hole was modeled, and Helius:MCT was used to predict progressive fiber and matrix failure caused by a displacement controlled load. Three different procedures specific to Helius:MCT were used:

1. In section 2.3, the Helius:MCT GUI was used to generate a user material.
2. In section 2.9, solution controls were defined.
3. In section 2.10, state variable (SVAR) output to the results file (.rst) was requested.

After the solution was completed, ANSYS was used to plot fiber and matrix failure in individual plies and to view the progression of failure within ply 2.